COMBUSTOR LINER TEMPERATURE PREDICTION: A PRELIMINARY TOOL DEVELOPMENT AND ITS APPLICATION ON EFFUSION COOLING SYSTEMS

A. Andreini, C. Carcasci, A. Ceccherini, B. Facchini, M. Surace
Dipartimento di Energetica "Sergio Stecco"
Via Santa Marta, 3
50139 Firenze - ITALY
Email: antonio.andreini@htc.de.unifi.it

D. Coutandin, S. Gori, A. Peschiulli
Avio S.P.A.
10040 Rivalta di Torino - ITALY
Email: daniele.coutandin@aviogroup.com

ABSTRACT

A reliable thermal design of a combustion chamber is a vital task to improve actual capabilities and accomplish the desired compromise among the distinct exigencies arising from modern devices. This paper presents the development of a design tool for temperature evaluation in combustor liners, by means of engineering simplified models and empirical correlations, coupled with a well-proven flow solver. Specific steps are provided to correctly take into account heat loads on both liner sides, local heat sink effects of cooling holes, coolant and hot gas flow path with particular attention to air extraction from the annulus - through primary, diluition and effusion holes - and its mixing with flame and exhausts. Combined convective and radiative heat transfer, as well as conduction and cooling issues, are then treated in details to perform several numerical simulations of different effusion cooling systems, evidencing the possible performance improvements of such arrangements in terms of metal temperature and coolant mass flow rate savings. Achievements and design practice are being exploited within the European Project NEWAC (NEW Aeroengine Core Concepts) to address Avio-PERM (Partially Evaporating Rapid Mixing) single annular combustor design.

OVERVIEW

Over the last ten years, there have been significant technological advances towards the reduction of emissions, strongly aimed at meeting the strict legislation requirements. Some very encouraging results have already been obtained but the achieved solutions have created other technical problems.

The aim to reach very low emission limits has recently changed several aspects of combustor fluid dynamics. Among them, combustor cooling experienced significant design efforts to obtain good performances with unfavorable boundary conditions.

Modern aeroengine combustors, mainly LPP-DLN, operate with premixed flames and very lean mixtures, i.e. primary zone air amount grows significantly, while liner cooling air has to be decreased. Consequently, important attention must be paid in the appropriate design of liner cooling system; in addition, further goals need to be taken into account: reaction quenching due to cool air sudden mixing should be accurately avoided, whilst temperature distribution has to reach the desired levels in terms of both pattern factor and profile factor (see Lefebvre [1]).

In recent years, the improvement of drilling capability has allowed to perform a large amount of extremely small cylindrical holes, whose application is commonly referred as effusion cooling. Even if this solution determines, at least in early part of the liner, a slight reduction of wall protection with respect to film cooling, the most interesting aspect is the significant effect of wall cooling due to the heat removed by the passage of coolant inside the holes (Gustafsson [2]). In fact, a higher number of small holes, uniformly distributed over the whole surface, permits a significant improvement in lowering wall temperature. From this point of view, effusion can be seen as an approximation of transpiration cooling by porous wall means, with a slight decrease in performances but without the same structural disadvantages. Nevertheless the high reduction of available coolant mass flow required to satisfy LPP-DLN flame dilution drives to test additional techniques to improve cold-side convective heat transfer. In this work two different devices were analyzed: the adoption of ribs as turbulence promoters on the cold-side surface of the liner, and the introduction of jets impinging on cooled surface.

NOMENCLATURE

c_p	Specific heat	$[J\,kg^{-1}K^{-1}]$
L_f	Luminosity Factor	$[-]$
l_b	Mean Beam Length	$[m]$
D	Effusion or impingement hole diameter	$[m]$
FAR	Fuel Air Ratio by mass	$[-]$
H	Annulus height	$[m]$
P	Absolute pressure	$[Pa]$
p	Partial pressure	$[Pa]$
T	Temperature	$[K]$
x	Streamwise pitch	$[m]$
y	Spanwise pitch	$[m]$
W	Plate width	$[m]$

Greeks

α	Absorbtion	$[-]$
σ	Boltzmann constant	$[WK^{-4}m^{-2}]$
θ	Effusion hole angle	$[^\circ]$
ε	Emissivity	$[-]$

Subscripts

0	Total quantity
g	Hot gas
s	Soot
T	Total emissivity
w	Wall
imp	impingement

Superscripts

$\overline{\square}$	Nominal values

1 CODE FEATURES

In this section the numerical tool adopted to calculate the performances of investigated cooling geometries will be presented. The study aims at evaluating as much as possible combustor cooling systems design parameters, such as:

- effusion holes spacing, angle, diameter;
- impingement holes pitches, gap, diameter;
- ribs dimension, spacing, annulus height;
- boundary conditions;

so as to determine their influence over external and internal metal temperature, coolant mass flow rate and pressure drop.

Heat conduction inside the metal and then its temperature distribution is obtained using a 1-D Finite Difference Model (FDM) of the liner with a in-house code, built-in inside the whole procedure, and then calculating metal conduction as far as hot and cold side boundary conditions are known. Hot side heat loads (both convective and radiative) are modelled in details as specified in the next section.

The analysis of the coolant fluid network is performed using a one-dimensional steady code, SRBC (Stator Rotor Blade Cooling), developed in-house during industrial research projects and successfully used in several works (Carcasci et al. [3], Carcasci and Facchini [4], Facchini et al. [5, 6], Arcangeli et al. [7]). The cooling system consists of a fluid network connecting one basic component to another, each one representing a particular region of the cooling system. In the investigated cases each simple model could represents a single of multiple effusion cooling row, impingement row or ribbed wall. Specification of geometric characteristics of single components, such as holes diameter, pitch, length, roughness, inclination angle and so on, can be custom selected. Coolant is considered a perfect gas subjected to wall friction and heat transfer, and flow field is solved in subsonic regime, using correlations to determine HTC, friction factor, cooling effectiveness. The user can specify boundary conditions for the fluid network in terms of inlet and outlet pressure or mass flow rate, depending on design specifications. Therefore SRBC, solving the fluid network, provides coolant side thermal boundary conditions for thermal calculation, in particular heat sink effect of coolant holes and effusion cooling effectiveness. Heat sink effect is evaluated by applying to the FDM model, for each row, heat removal given by the mean heat transfer coefficient and adiabatic wall temperature inside the holes. Film cooling effectiveness is provided by SRBC using the correlation in L'Ecuyer and Soechting [8], which was derived in 1985 after an extensive search in the literature performed to assemble a comprehensive bibliography of film cooling (384 publications from 1964-1983) and to select the primary sources of data to be used in the development of a reliable flat plate adiabatic effectiveness correlation. It calculates the adiabatic film cooling effectiveness for each row (once the blowing ratio, velocity ratio and row geometry are known), and then the principle proposed by Sellers and presented in Lakshminarayana [9] is used to superimpose the effects of each row. Achieved film cooling effectiveness is then used together with flame temperature and outlet coolant temperature to evaluate external adiabatic wall temperature. Since the mass flow rate and pressure drop is wall temperature dependent, an iterative procedure is required. Convergence is achieved when differences in pressures and mass flow remain unchanged or below an error set by the user (0.01% in this case). SRBC code can also be used without the thermal FEM model, to perform fixed metal temperature or adiabatic calculations on fluid networks.

As declared in the last paragraph two additional convective heat transfer mechanisms were tested: ribs on cold-side surface and impingement. Both techniques were applied leaving unchanged the standard effusion drilling of the liner: classic effects superposition was employed. Heat transfer augmentation due to ribs were estimated with a correlation due to Han [10], while impingment heat transfer was computed with Florschuetz formulation [11].

1.1 Heat loads

The methodology here followed to perform a preliminary assessment of the heat loads on liner walls is based on the classical one-dimensional approach suggested by Lefebvre [12], [13], [14] and later improved by Kretschmer and Odgers [15] and P. Gosselin and Kretschmer [16].

As conceptually described in fig. 1 the procedure is based on the balance among different mechanisms of heat transfer:

- R_1 internal radiation
- C_1 internal convection
- R_2 external radiation
- C_1 external convection
- K_{1-2} conduction within liner metal

Energy balance in steady state condition implies that (neglecting heat conduction in axial direction):

$$R_1 + C_1 = R_2 + C_2 = K_{1-2} \tag{1}$$

Such equation have to be solved at different axial locations of

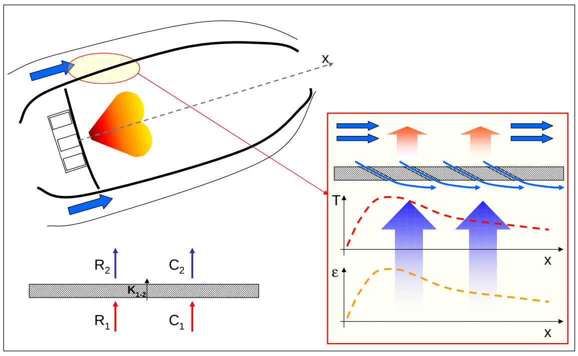

Figure 1. Concept view of one dimensional thermal design methodology

combustor liner starting from one dimensional distribution of main flow variables computed by SRBC code. Radiative heat flux on liner hot wall (R_1) can be computed recalling the compact equation suggested by Lefebvre [14]:

$$R_1 = 0.5 \cdot \sigma \cdot (1+\varepsilon_w) \cdot \varepsilon_g T_g^{1.5} \left(T_g^{2.5} - T_w^{2.5}\right) \tag{2}$$

Which is a simplification of standard radiative heat flux formula having assumed that between gas emissivity ε_g and absorption α_g holds the following relationship:

$$\frac{\alpha_g}{\varepsilon_g} = \left(\frac{T_g}{T_w}\right)^{1.5} \tag{3}$$

The overall accuracy is therefore demanded to the accuracy and reliability of predicted gas and wall emissivity. Liner walls emissivity can usually be safely estimated taking into account material properties, surface roughness and temperature (typical values between 0.7 and 0.8). Prediction of a global value for gas emissivity is a more complex task.. The emission and absorption of radiation by non luminous fossil fuel combustion products is due mainly to transitions between vibrational and rotational energy states of the polyatomic gases CO_2 and H_2O. These gases are highly non-grey and emit and absorb radiation at spectral bands centred around specific wavelengths. A proper evaluation of gas emissivity requires the computation of the contribution of each gases accordingly to its wavelengths. Nevertheless one of the main issue is even the reduced set of input data available in the early phase of design process. It's therefore usually neglected the use of formulations which allow to take into account different spectral behaviour of constitutive gases (as for instance the WSGGM - Weighted Sum of Grey Gases Model [17]), which would require the knowledge of mixture composition along the liner. The formulation is greatly simplified if the gas is treated as a mixture of grey gases. With this approach total emissivity is usually expressed in the form:

$$\varepsilon_g = f\left(P, p_k, l_b, T_g\right) \tag{4}$$

being p_k the partial pressures of constitutive gases, P the absolute pressure, l_b the mean beam length and T_g the gas temperature. Another common issue is the luminous behaviour of the flame mainly due to the presence of soot. A proper computation of soot emissivity would require the exact distribution of soot volume fraction along the liner, which is not easy to be estimated even with empirical correlations. A widely used expression to predict gas emissivity is the well known correlation by Reeves [18]:

$$\varepsilon_g = 1 - e^{\left[-290 \cdot P \cdot (FAR \cdot l_b)^{0.5} \cdot T_g^{-1.5}\right]} \tag{5}$$

which is valid only for pressure below 5 bar. An extension to Reeves correlation was suggested by Lefebvre in order to account for luminous radiation. The model is based on the introduction of the concept of Luminosity Factor L_f. It is an empirical correction to non luminous emissivity which is largely dependent on the carbon/hydrogen ratio of the fuel and to a lesser extent to combustor pressure. Lefebvre correction to Reeves correlation is:

$$\varepsilon_g = 1 - e^{\left[-290 \cdot P \cdot L_f \cdot (FAR \cdot l_b)^{0.5} \cdot T_g^{-1.5}\right]} \tag{6}$$

Updated formulas to compute L_f, valid for modern engines burning kerosene fuels are:

$$\begin{aligned} L_f &= 3.0 \cdot \left(C/H - 5.2\right)^{0.75} \\ L_f &= 336/\left(\%H_2\right)^2 \end{aligned} \tag{7}$$

being C/H the carbon over hydrogen mass ratio and $\%H_2$ the percentage by mass of hydrogen in the fuel. The Reeves-Lefebvre correlation tends to overestimate gas emissivity when used for absolute pressure beyond 5 bar (great overestimation for $P > 10$ bar). Another limit is the strict validity only for lean flames: for rich flames it should be used limiting FAR to the stoichiometric value. For high pressure cases (which are typical at max take off conditions where thermal design of combustor is verified) a valid alternative to previous expressions is the empirical formula proposed by Lefebvre and Herbert [12] and based on the experimental data set of Farag [19]. The expression is:

$$\varepsilon_g = \left(0.15 - 0.00005 \cdot T_g\right)\left(2 \cdot P \cdot FAR \cdot l_b\right)^{\left(0.20 + 0.00015 T_g\right)} \tag{8}$$

This correlation is strictly valid only for:

- $P < 40$ bar
- $1200 < T_g < 2400$K
- $FAR < FAR_{st}$

Also in this case when used for rich mixture it should be set $FAR = FAR_{st}$. In order to be able to use more accurate expressions for non luminous flame emissivity leaving the possibility to take into account luminous radiation without having to know soot volume fraction, a generalized formulation of Lefebvre's Luminosity Factor was introduced:

$$\varepsilon_T = 1 - (1 - \varepsilon_g)^{L_f} \tag{9}$$

being ε_g non luminous gas emissivity computed with whatever correlation.

Hot side convection (C_1) was evaluated following classical Lefebvre suggestions [1]. Particular attention was paid to take into account the effect of tangential velocity induced by the swirler in the Primary Zone, which acts to locally increase heat transfer coefficient. Local increase in heat transfer due to high momentum film cooling in the early region of liner was considered as well. Concerning external radiation no particular arrangements were adopted with respect to classical theories. As described in the previous paragraphs, external convection is computed by the 1-D flow solver SRBC.

1.2 Overall procedure flow chart

In order to have a more precise comprehension of the overall procedure adopted and to better describe the connections among the different tools on which it is based, a schematic flow chart is reported in fig. 2.

1.3 Methodology validation

In order to asses the accuracy and the reliability of the implemented methodology, a validation test case was selected. The considered geometry is described in detail in [20]: it is an effusion cooled flat plate for which experimental measurements of overall effectiveness are available. In the work of A. Andreini [20] and coworkers, experiments are compared with results obtained with a calculation methodology which differs from the one here described only for the approach used to solve metal conduction (2-D FEM commercial solver).

Calculations have been repeated using the present approach, where a 1-D strategy is used to solve metal conduction. Results are reported in figure 3. It's interesting to point out that in the central region of the domain, where effects related to plate boundaries are small, the accuracy of the new methodology is equivalent to the 2-D study with an overall good agreement with measured data.

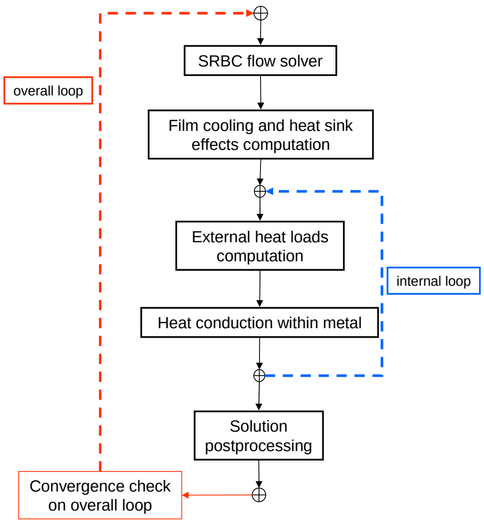

Figure 2. Overall procedure flow chart

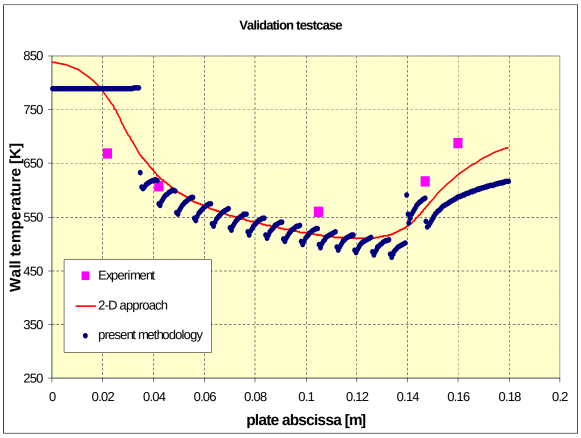

Figure 3. Results of validation testcase

2 COMBUSTOR GEOMETRY AND INVESTIGATED SOLUTIONS

Analysis began choosing a reference case. It was selected as a typical film cooled inner liner for diffusive aeronautic combustors. It is provided with an upstream cooling slot, a typical effusion holes pattern with a nominal holes diameter $\bar{D}$ below $1.0\,mm$, a nominal inclination angle $\bar{\theta}$ between 30° 45°, no Thermal Barrier Coating and a nominal metal thickness $\bar{W}$ around $1.0\,mm$.

Hot gas temperature distribution, representing a realistic diffusion flame behavior, is shown in fig. 4.

Results of reference thermal analysis are plotted in fig. 5 in terms of hot side metal temperature vs. linear abscissa along the

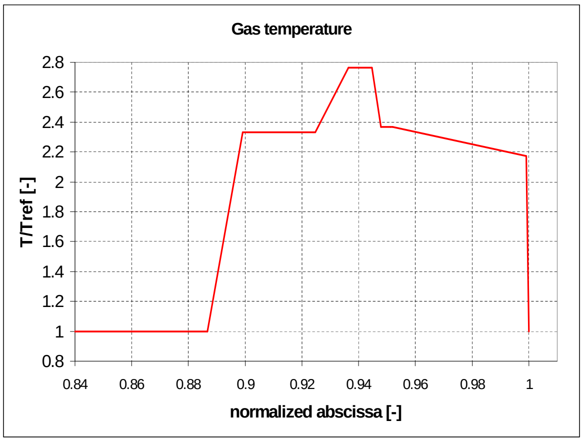

Figure 4. Hot gas temperature distribution

liner.

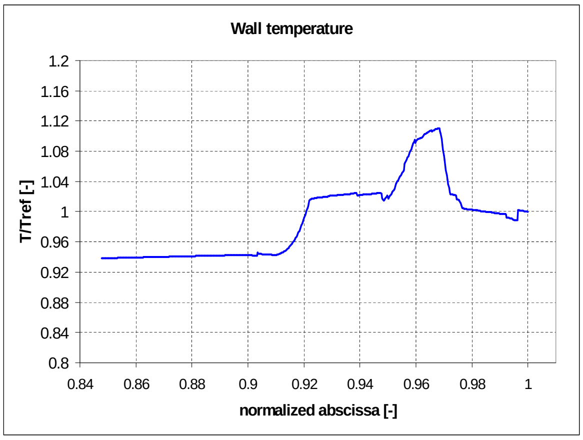

Figure 5. Base case gas side metal temperature distribution

Then analysis proceeded performing calculation on the same geometry but varying effusion cooling holes diameter and angle and TBC thickness. All tests were performed keeping unchanged the pressure drop across the liner (around 5%). The main effects selected to be investigated were: effusion cooling holes diameter and angle, decrease in total coolant passage area (keeping the same streamwise and spanwise pitch aspect ratio) and TBC presence.

Different cases will be now listed in details.

1. **30**: as previously described: upstream slot, 3 cases with 3 different diameters ($D = \overline{D},\ 0.75\overline{D},\ 0.5\overline{D}$), nominal angle $\overline{\theta}$ and metal thickness, no TBC
2. **30TBC**: as **30**, but with a layer of thermal barrier coating having a thickness of $0.2\overline{W}$ ($D = 0.75\overline{D}$ not studied)
3. **30X1.2**: as **30**, but with streamwise and spanwise pitch increased of 20%, i.e. hole area decreased of 31%
4. **30X1.2TBC**: as **30X1.2**, but with $0.2\overline{W}$ of thermal barrier coating
5. **30X1.5**: as **30**, but with streamwise and spanwise pitch increased of 50%, i.e. hole area decreased of 56%
6. **30X1.5TBC**: as **30X1.5**, but with $0.2\overline{W}$ of thermal barrier coating
7. **15**: as **30**, but with $\theta = 0.5\overline{\theta}$, 3 cases with 3 different diameters ($D = \overline{D},\ 0.75\overline{D},\ 0.5\overline{D}$)
8. **15X1.2**: as **15**, but with streamwise and spanwise pitch increased of 20%, i.e. hole area decreased of 31%
9. **15X1.5**: as **15**, but with streamwise and spanwise pitch increased of 50%, i.e. hole area decreased of 56%

Last step of analysis concerns the introduction of compound impingement-effusion and rib-effusion cooling systems. Geometrical configurations for this systems were chosen with the aim of maximize heat transfer from the liner to the coolant and are reported in fig. 6 and 7 respectively. Impingement was designed introducing a perforated plate, with $D = 5/3\overline{D}$ diameter holes, 3 mm far from the liner outer wall. Each impingement hole was thought to provide the same coolant mass flow of an effusion one therefore no cross flow on the following was modeled. Analysis on rib-effusion compound system were carried out designing two configurations:the first having the same gap height of the original one, the second with a gap height reduced to $1/4$. In both the configurations the same rib geometrical parameters were assumed.

Compounds systems were calculated always taking as reference the base case (holes diameters $D = \overline{D}$ and $\theta = \overline{\theta}$) and the corresponding ones with increased pitches (X1.2 and X1.5).

3 RESULTS

Obtained results are shown in fig. 8, in terms of average metal temperature vs. coolant mass flow rate. Different colors represent different cases and the circle dimension is proportional to the hole diameter ($D = 0.5\overline{D}$ the smallest, $D = \overline{D}$ the largest). Some considerations can be drawn by looking at the figure:

Not being a novel aspect, it's however remarkable that a smaller diameter allows a decrease in both mass flow rate and average metal temperature.

Similarly, a lower injection angle, positively affecting the heat sink effect and the adiabatic effectiveness, causes a decrease in both mass flow rate and average metal temperature.

TBC is able to lower metal temperature of about $10-15\,K$, with limited mass flow rate variations.

Combining the reduction in hole area and the previous beneficial effects, it appears to be theoretically achievable a sav-

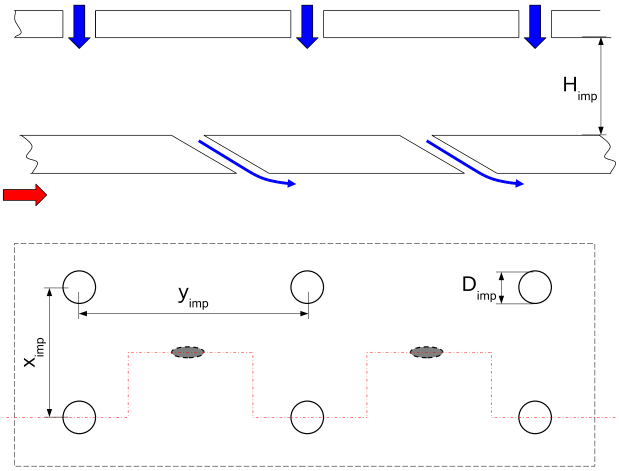

Figure 6. Compound impingement-effusion geometry

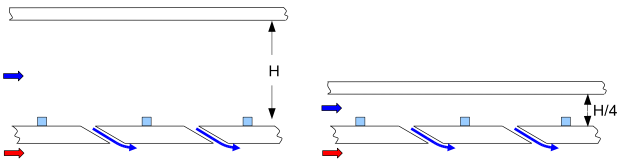

Figure 7. Compound rib-effusion geometry

ing of 70% of coolant mass flow rate with the same reference metal temperature.

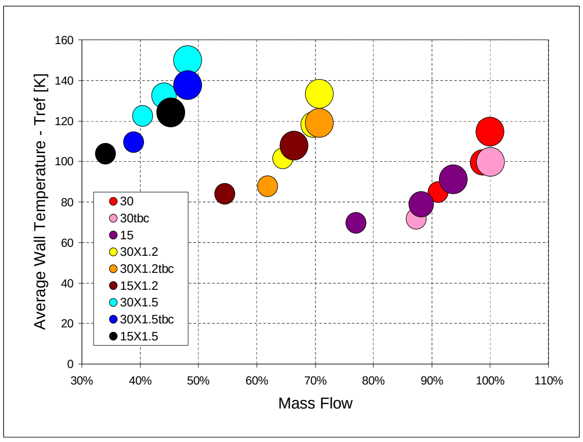

Figure 8. Simulation results

Somehow different results could be achieved by varying the hole pattern aspect ratio, i.e. the streamwise and spanwise pitch aspect ratio, as its value influences both the heat sink effect and the adiabatic effectiveness trend.

In fig. 9 results for all compounds systems are reported in terms of reduced coolant mass flow versus average liner wall temperature and compared to the base cases and to the best results achieved varying effusion holes parameters ($D = 0.5\bar{D}$ and $\theta = 0.5\bar{\theta}$).

Impingement: Results show the possibility to obtain good cooling performances by using this type of compound. Reduction in coolant mass flow as result from calculations is mainly due to the presence of the impingement perforated plate which increase the pressure drops. Similar temperature level are so achievable with a reduced coolant mass flow (about 65% of the base one).

Improvement in cooling performances are similar for all of the three cases calculated suggesting the possibility to reduce holes number and to introducing impingement as one of the way of obtaining the same liner temperature level with a reduce coolant needing.

Ribs: In the cases having the original annulus hight presence of ribs on surface does not affect coolant mass flow rate due to the few pressure losses added to the original configuration. Average liner temperature is slightly decreased because of the effect of enhanced heat transfer coefficient on the coolant side. Great advantages seem to be achievable adopting a reduced hight annulus. Increase in pressure drop causes a lower average pressure in the coolant channel and a decrease in coolant mass flow. Higher velocity in the annulus increase heat transfer coefficient on ribbed surface. A big coolant flow saving can be achieved in this way and the effect seems to be increased reducing the number of the holes.

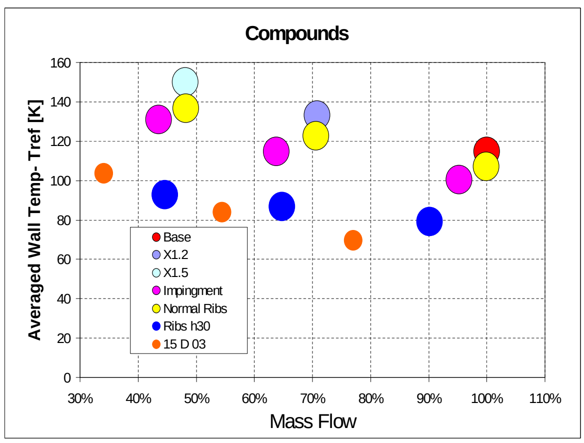

Figure 9. Compound system results

It has to be noted that calculation with ribs were obtained assuming a fixed temperature casing which in these calculation played an important rule maintaining the coolant at very low temperature. Another important assumption deals with a zero effect of ribs and impinging jets presence on effusion system behavior which is the common approach followed in a 1-D fluid network analysis (effects superposition).

CONCLUSIONS

The present study shows a simple and fast calculation tool for the complete thermal design and analysis of combustor's liner. Specific calculations have been performed on a actual liner geometry to test different effusion cooling solutions with or without cold-side heat transfer augmentation via ribs or impingement.

Results point out the relevant positive effect on effusion cooling performances of a reduction of holes' diameter and drilling angle.

Combined cooling systems (effusion with ribs or impingement) seem to give interesting results but their real efficiency should be verified when the interaction with effusion holes have been investigated through CFD calculations or experiments. This last researches will be the subject of future works.

ACKNOWLEDGMENT

The present work was supported by the European Commission as part of FP6 IP NEWAC research program, which is gratefully acknowledged together with consortium partners.

References

[1] A. H. Lefebvre. *Gas Turbine Combustion*. Taylor & Francis, 1998.

[2] K. M. B. Gustafsson. Experimental studies of effusion cooling. *Chalmers University of technology, Department of Thermo and Fluid Dynamics*, (`www.tfd.chalmers.se/~lada/postscript_files/bernhard_phd.pdf`), 2001.

[3] C. Carcasci, B. Facchini, and G. Ferrara. A rotor blade cooling design method for heavy duty gas turbine applications. *ASME Cogen-Turbo Power*, (95-CTP-90), 1995.

[4] C. Carcasci and B. Facchini. A numerical procedure to design internal cooling of gas turbine stator blades. *Revue Générale de Thermique*, **35**, 1996.

[5] B. Facchini, M. Surace, and S. Zecchi. A new concept of impingement cooling for gas turbine hot parts and its influence on plant performance. *ASME Turbo Expo*, (GT-2003-38166), 2003.

[6] B. Facchini, M. Surace, and L. Tarchi. Impingement cooling for modern combustors: experimental analysis and preliminary design. *ASME Turbo Expo*, (GT2005-68361), 2005.

[7] L. Arcangeli, B. Facchini, M. Surace, and L. Tarchi. Correlative analysis of effusion cooling systems. *ASME Journal of Turbomachinery*, **129**, October, 2007.

[8] M. R. L'Ecuyer and F. O. Soechting. A model for correlating flat plate film cooling effectiveness for rows of round holes. In *AGARD Heat Transfer and Cooling in Gas Turbines 12p (SEE N86-29823 21-07)*. September 1985. Provided by the Smithsonian/NASA Astrophysics Data System `http://adsabs.harvard.edu/cgi-bin/nph-bib_query?bibcode=1985htcg.agarQ....L&db_key=PHY`.

[9] B. Lakshminarayana. *Fluid Dynamics and Heat Transfer of Turbomachinery*. John Wiley & Sons, Inc., 1996.

[10] J. Han, S. Dutta, and S. Ekkad. *Gas Turbine Heat Transfer and Cooling Technology*, pages 251–287. Taylor & Francis, 2000.

[11] L.W. Florschuetz, C.R. Truman, and D.E. Metzger. Streamwise flow and heat transfer distributions for jet array impingement with crossflow. *ASME Journal of Heat transfer*, 103:337–342, 1981.

[12] Arthur H. Lefebvre and M. V. Herbert. Heat-transfer processes in gas turbine combustion chambers. *Proc. Instn. Mech. Engrs.*, 174:463–473, 1960.

[13] Arthur H. Lefebvre and E. R. Norster. The influence of fuel preparation and oparating conditions on flame radiation in a gas turbine combustor. *ASME*, (72-WA/HT-26):463–473, 1972.

[14] Arthur H. Lefebvre. Flame radiation in gas turbine combustion chambers. *Int. Journal of Heat and Mass Transfer*, 29:1493–1510, 1984.

[15] D. Kretschmer and J. Odgers. A simple method for the prediction of wall temperatures in a gas turbine combustor. *ASME*, (78-GT-90), 1972.

[16] A. de Champlain P. Gosselin and D. Kretschmer. Prediction of wall heat transfer for a gas turbine combustor. *Proc. Instn. Mech. Engrs.*, 213:169–180, 1999.

[17] A. Sayre N. Lallemant and R. Weber. Evaluation of emissivity correlations for h_2o co_2 n_2 air mixtures and coupling with solution methods of the radiative transfer equation. *Progr. Energy Combust. Sci.*, 22:543–574, 1997.

[18] D. Reeves. Flame radiation in an industrial gas turbine combustion chamber. *National Gas Turbine Establishment*, NGTE-Memo-285, 1956.

[19] I. H. Farag. Nonluminous gas radiation: approximate emissivity models. *Proc. 7th Int. Heat Transfer Conf.*, R6:487–492, 1982.

[20] B. Facchini E. Mercier M. Surace A. Andreini, J.L. Champion. Advanced liner cooling numerical analysis for low emission combustors. *25th International congress of the aeronautical sciences*, 2006.